

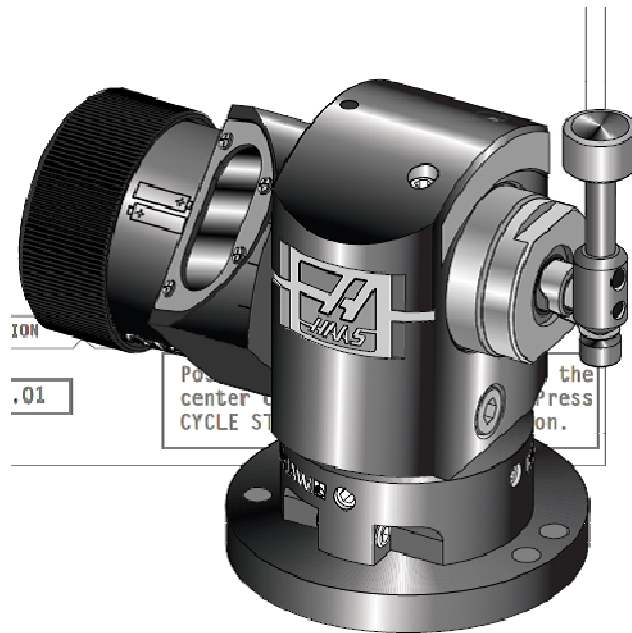


Haas Factory Outlet

A Division of Productivity Inc

HFOMN/Haas CNC Series

WIPS (Wireless Intuitive Programming) *and VQCPS (Visual Quick Code)* *Probe Training Manual*



This Manual is the Property of Productivity Inc

*The document may not be reproduced without the express written permission of
Productivity Inc.*

*The content must not be altered, nor may the Productivity Inc name be removed
from the materials.*

*This material is to be used as a guide to operation of the machine tool. The
Operator is responsible for following Safety Procedures as outlined by their
instructor or manufacturer's specifications.*

To obtain permission, please contact trainingmn@productivity.com.


WIPS/VQCPS Probe Training Manual

Table of Contents

INTRODUCTION TO WIPS/VQCPS PROBE TRAINING	3
ELEMENTS OF THE MACHINE PROBE SYSTEM	4
OMI – OPTICAL MACHINE INTERFACE	4
OMP – OPTICAL MACHINE PROBE	5
OTS – OPTICAL TOOL SETTER	6
BATTERIES.....	8
PROBE TEST	10
PROBE PROGRAMS.....	11
MACRO PROGRAM LIST.....	12
PROBE ALIGNMENT.....	14
VQCPS – VISUAL QUICK CODE PROBE SYSTEM	15
PROBE CALIBRATION	15
SETTING OF TOOL LENGTH OFFSETS USING VCQ	20
SETTING WORK OFFSETS USING VISUAL QUICK CODE	24
WIPS – WIRELESS INTUITIVE PROBE SYSTEM.....	27
PROBE CALIBRATION	27
SETTING OF TOOL LENGTH OFFSETS USING IPS.....	30
SETTING OF WORK USING IPS	35



For more information on Additional Training Opportunities
or our Classroom Schedule
Contact the Productivity Inc Applications Department in Minneapolis:

 763.476.8600

Visit us on the Web: www.productivity.com

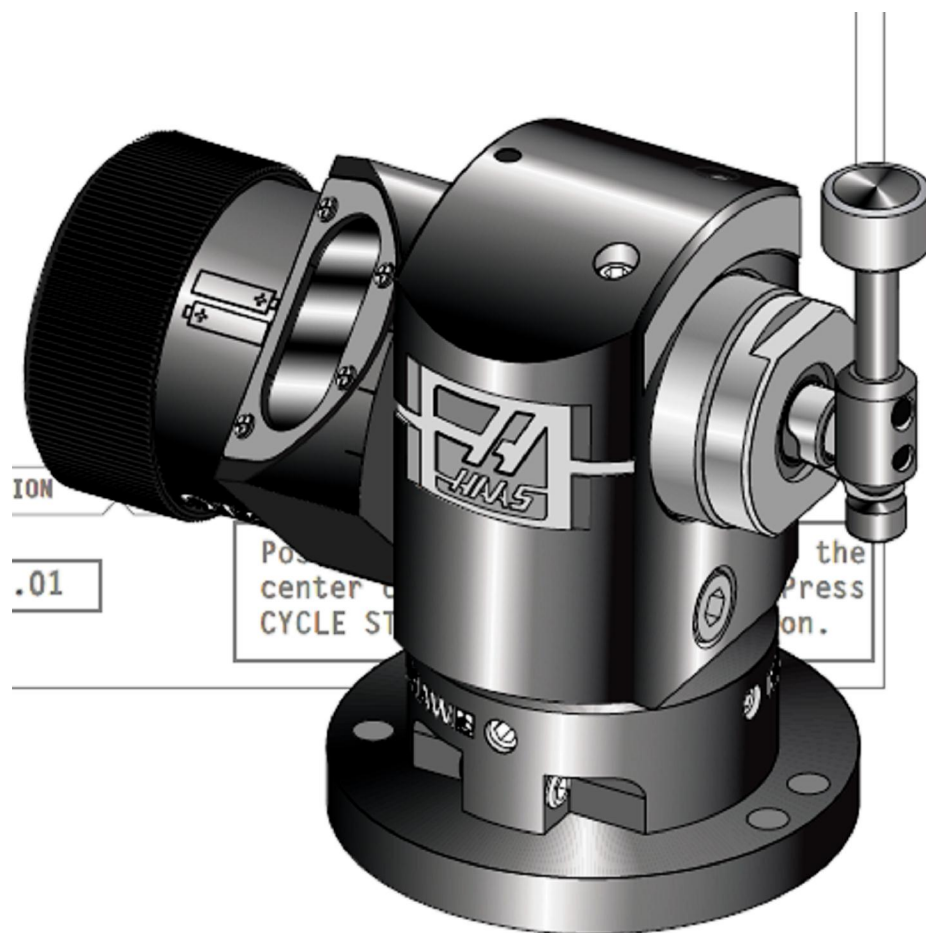
Click on the Training Registration Button

 trainingmn@productivity.com

Introduction to WIPS/VQCPS Probe Training

This class is designed to introduce new users of machine tool probes how to set Tool Length Offsets and Work Offsets using VQC (Visual Quick Code) and WIPS (Wireless Intuitive Probe System) software. The course does not cover how probe systems may be used in cycle gauging, inspection or automatic offset updates. This course is designed just to teach how WIPS and VQC may be used to simplify and speed set up of the Haas vertical machining center.

This is achieved by a predictive process control of tool setting and part setting. With probe systems the length of a tool from spindle gauge-line may easily determined. Also accurate diameter of spinning tools may be set. Various spindle probe routines determine different types of datum used to establish work co-ordinate systems. Probe routines may be also run to detect broken tools.

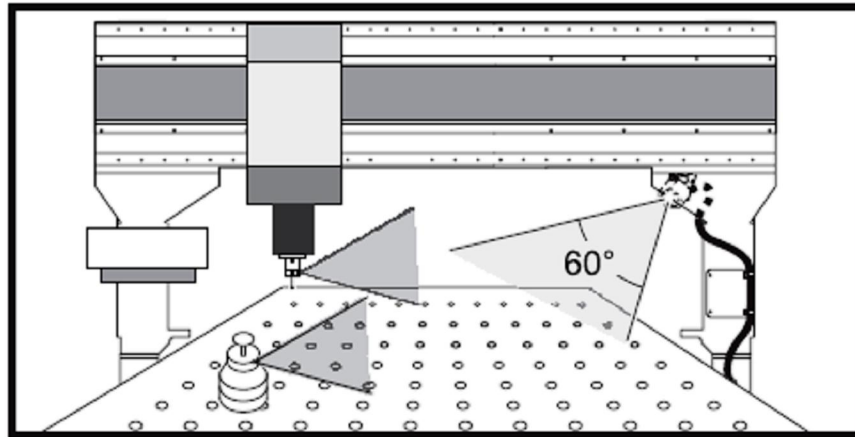


Elements of the Machine Probe System

The machine probe system consist of three units: the OMI (Optical Machine Interface), the OTS (Optical Tool Setting) table probe and the OMP (Optical Machine Probe) spindle probe.

OMI – Optical Machine Interface

The **OMI (optical machine interface)** is tied in directly to the machine CNC control. The OMI or machine interface communicates with the other two probes by modulated infra red light. It acts as an optical receiver which sends and receives signals from the other two probes directly to the CNC machine control. Below shows the optical Machine Interface on the right which can communicate with either the table or the spindle probe.



The table probe, spindle probe and the machine interface use LEDs and photo diodes to communicate back and forth with each other. In each of the devices there are specific LEDs which indicate probe status, battery status and any error conditions. The machine interface power supply comes from the machine control while the table and spindle probes require batteries.

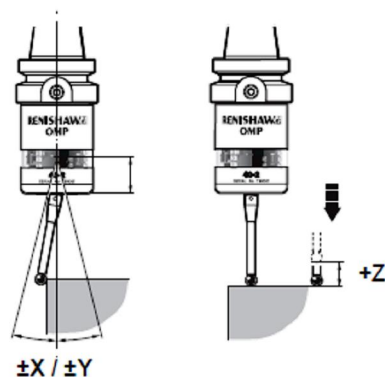
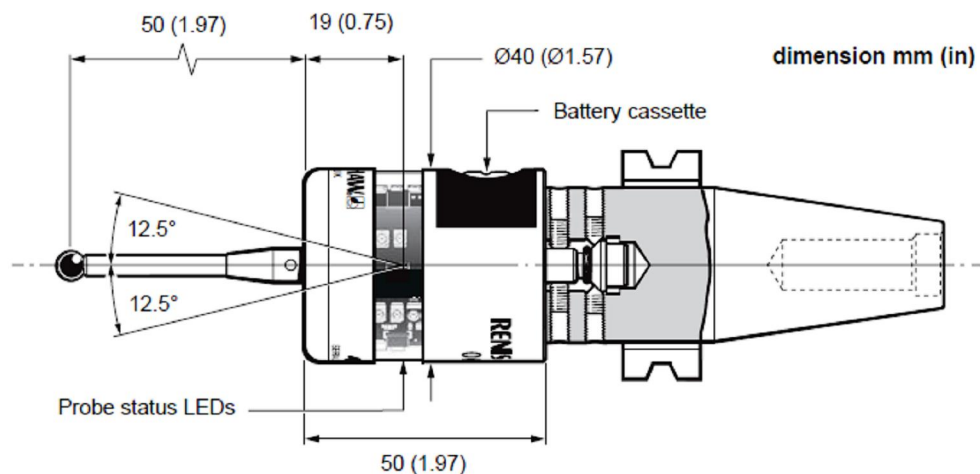
OMP – Optical Machine Probe

The **OMP** or **optical machine probe** is a 3D touch triggered inspection probe loaded in a tool holder. It is used to set up work offsets and inspection of parts in a Haas machining center. It will sense direction hits in 5 different directions +/-X, +/-Y, and +Z. Because of the sensitive electronics it is characterized as a **heavy tool** in the **Tool Pot Position page** so that it will only index at 25% rapid. Normally it is designated as the last tool in the side mount carousel or larger. With a 24 tool holder machine it may be designated as T25.

The OMP operates most of the time in the standby mode. It is activated by a **switch on** signal from the optical machine interface to the operating mode. After it is done with a probing routine it is switched off by a signal from the optical machine interface or pressing the RESET key.

The Renishaw probe comes with a standard 50 mm long ceramic stylus. At the end is a 6mm ruby ball. As the ceramic will easily break a (break away pin) is not required with this type of stylus. Other length probes with different diameter balls and lengths are available and may be used.

Renishaw Probe OMP40-2 Dimensions
(From Renishaw *Data Sheet H-5514-8200-01-A*)



Stylus overtravel limits		
Stylus length	±X / ±Y	+Z
50 (1.97)	12 (0.47)	6 (0.24)
100 (3.94)	22 (0.87)	6 (0.24)

OTS – Optical Tool Setter

The **OTS, Optical Tool Setter or table probe** is used to set tool length offsets, measure the diameters of tools or detect broken tools. The table probe gives precise measurements of tool length and diameter. It is similar to the spindle probe as it is run by batteries and is turned on and off by the OMI or machine interface. With the Renishaw OTS a weak link break pin P/N A-5003-5171 is between the stylus and the probe body. This protects the probe body from over travel or a collision.

The Tool Setting probe is normally mounted on the table of the vertical machine center. Haas recommends placing the probe on the right upper hand corner of the table. In this configuration the probe window would face away from any flying chips and opposite the tool changer side. The Probe base is held down by a socket head cap screw and T-nut. The Renishaw probe is mounted on the base using a locating pin for proper alignment. The probe is fastened down by two set screws, see B below.

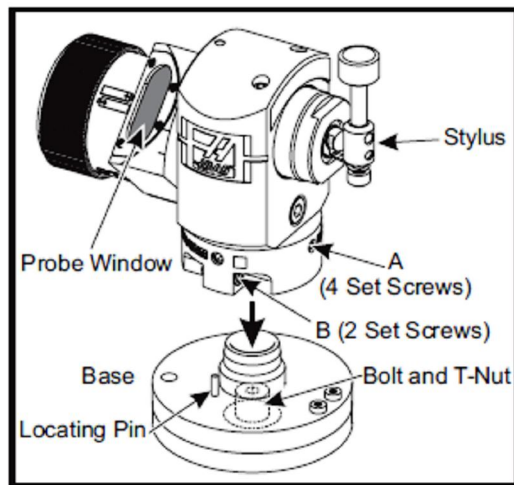
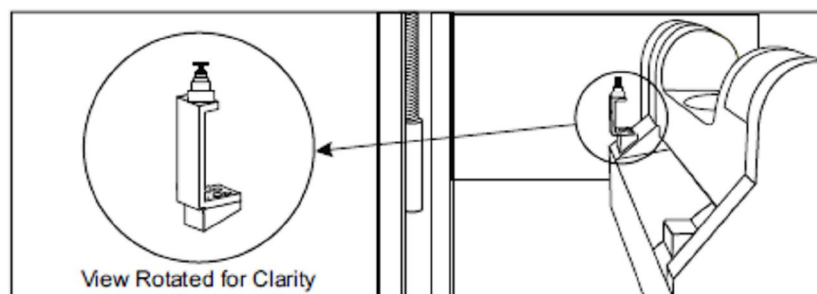


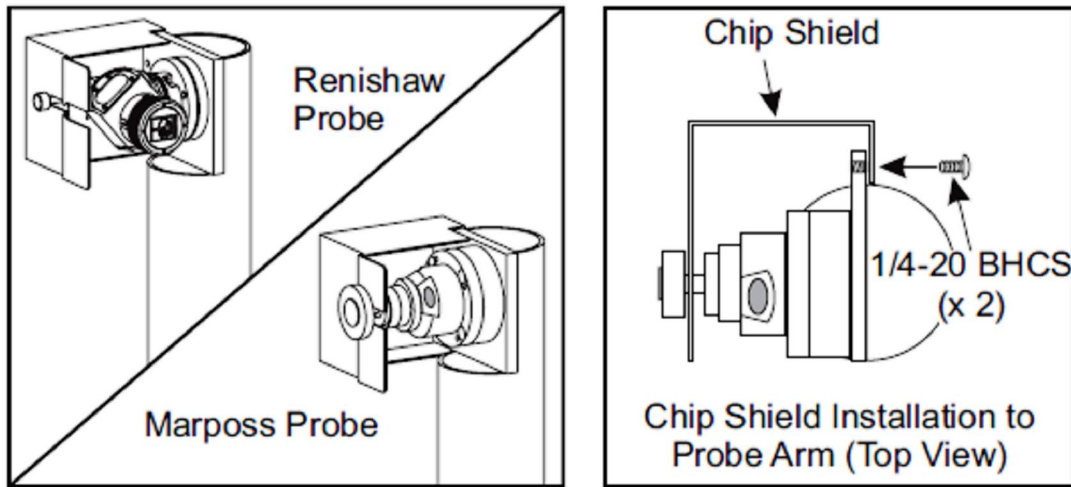
Table probes may also be mounted on plates. Two dowel pin holes are located on the base for mounting the probe on a plate where removal of the probe is required. This will relocate the probe in the same location.

Table probes may also be mounted on risers for particular applications. Special brackets are needed with trunion mills, see below:



VF-6TR Probe Mounting

On horizontal mills the probe is mounted horizontally. On horizontal machines special brackets or arms are needed. See below:



See Haas ES0583 Document on WIPS Quick Setup for details.

Renishaw and Marposs Replacement Part Numbers

Marposs

- 60-0041 - Ceramic Stylus (Spindle Probe)
- 60-0042 - Disk Stylus (Table Probe)
- 60-0043 - Break Away Pin (Table Probe)
- 60-0075 - Angle Stylus (Table Probe)

Renishaw

- 60-0026 - Ceramic Stylus (Spindle Probe)
- 60-0028 - Disk Stylus (Table Probe)
- 60-0029 - Stylus Holder (Table Probe)
- 60-0030 - Link Break Protect (Table Probe)
- 60-0034 - Extension (Table Probe)

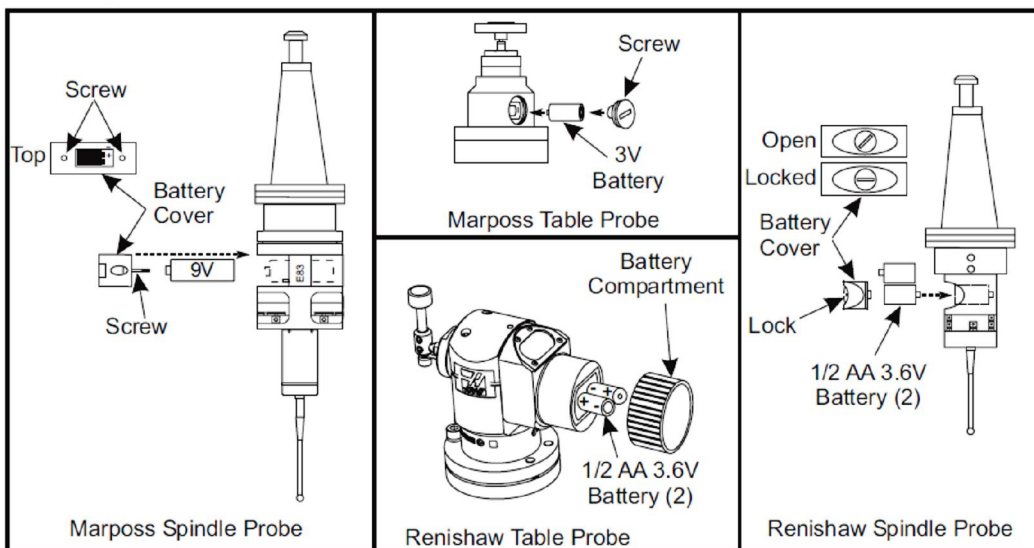
Batteries

WIPS table and spindle probes require batteries. The OMI or optical machine interface is supplied with power directly thru the machine controls. Below gives the information on the different batteries required for Marposs and Renishaw table and spindle. It also gives replacement information.

Batteries for Probing Options

			Haas Part #
Marposs	E83 Spindle Probe	9.0V Transistor	99-4354
Marposs	OTS30 Table Probe	3.0V CR123A Lithium	99-43554
Renishaw	OMP40 Spindle Probe OTS Table Probe	3.6V Lithium 1/2 AA	99-43553

Probe Battery Replacement



Marposs Spindle Probe – The Marposs Spindle Probe contains (1) 9-Volt battery. Use a 3mm Allen wrench to unscrew and remove the battery cover located on the side of the probe. Remove the spent 9V battery and insert a new one. Screw the battery cover back down on the 9V battery.

Marposs Table Probe – The Marposs Table Probe contains (1) 3-Volt battery. Use a coin to unscrew and remove the battery cover located on the side of the probe. Remove the spent 3V battery and insert a new one. Screw the batter cover back down on the 3V battery.

Renishaw Spindle Probe – The Renishaw Spindle Probe contains (2) 1/2 AA 3.6V batteries. Use a coin to unscrew and remove the battery cover located on the side of the probe. Remove both the spent 1/2 AA 3.6V batteries and insert new ones. Screw the batter cover back down on the 1/2 AA 3.6V batteries.

Renishaw Table Probe – The Renishaw Table Probe contains (2) 1/2 AA 3.6V batteries. Unscrew the battery cover/holder from the battery compartment located on the side of the probe. Remove both the spent 3.6V batteries and insert new ones into the battery holder. Place the battery cover/holder back onto the battery compartment.

(From Haas Wireless Intuitive Probe System, Quick Start Set-up and Operation. ES0583 rev C)

Battery life depends on how much the probe is used. The probe is only turned on when it is being used. After it is done with its routine it is shut off to conserve battery life. Renishaw gives the **standard life** of the LTC ½ AA 3.6 volt in the **OTS** as 300 hours continuous use, **5% usage** (72 minutes a day) at **140 days**. For the **OMP**, an average life of 170 hours continuous use, **5% usage** (72 minutes a day) at **115 days**. Most problems with probes stem from low batteries. Because the battery life of the probe on average is only about three months it is wise to keep spare batteries on hand. This is especially important if there is a night shift.

Trouble Shooting Battery Problems:

Most communication problems with IPS are caused by either low or dead batteries. Another problem may be accumulation of chips on the table probe. Looking at the LED lights from the probes will give an indication of weak or dead batteries.

LED Probe Status

Probe is in Operating Mode	Flashing Green light
Probe triggered in Operating Mode	Flashing Red light
Low battery in Operation Mode	Flashing Green and Blue
Probe triggered in Operating Mode low battery	Flashing Red and Blue
Battery Dead	Constant Red
Unsuitable Battery	Flashing Red Flashing Red and Green Flashing Red, Green and Blue

Probe Test

The following code verifies if the table probe (OTS), the spindle probe (OMP) and the machine interface (OMI) are functioning correctly.

When the vertical machine center is powered up the OMI **3 LED's** (light emitting diodes) are all **red**. The spindle and table probe are both off, no LEDs are on and black.

Enter the following code in MDI. Running the short program in MDI will **turn the table probe (OTS) on**.

```
M59 P1133 (M59 forces or sets an output relay)  
G04 P1.0  
M59 P1134
```

If the machine interface and the table probe are working properly the **table probe LED starts blinking green**. The machine interface or **OMI 3 LEDs will turn solid green**. Tapping the probe with your finger (simulating a hit) will temporarily turn the **table probe LED flashing red** and **one** of the **LED's** of the machine interface **OMI** simultaneously will temporarily **turn red**. Also a **beep** will be heard. Press the **RESET** button to turn off the table probe.

The following code will turn on the spindle probe.

```
M59 P1134  
G91 G31 X-1.0 F5.0
```

When the spindle probe (**OMP**) is turned on a LED in the spindle probe will start **blinking green**. At same time the machine interface or **OMI 3 LEDs will turn solid green**. The second line of code commands the machine to make an incremental move in the -X direction. Tapping the probe with your finger (simulating a hit) will stop the machine movement in X. This illustrates the G31 skip function. Along with stopping the machine movement the tapping will temporarily turn the **spindle probe LED flashing red** and **one** of the **LED's** of the machine interface **OMI** simultaneously will temporarily **turn red**. Also a **beep** will be heard. After running the above probe tests press the **RESET** button to turn off the different probes.

Probe Programs

Macro Programs: A **Macro** is a sub-program which includes commands that are not G code. Macros include decision statements such as IF, GOTO, various math functions and the use of variables. It provides more options for the use of CNC. Sometimes they are used to perform repetitive functions such as G83 peck drilling. G83 is a Macro program which is not visible on the Haas CNC control.

Probing macros direct the machine to perform complex probe movements. From these motions, probe contacts or hits are used to calculate offset locations, calculate distances, tool lengths with the use of variables. Most macro programs are long and difficult to follow simply by the fact of the use of so many different variables.

Variables: In computers a variable is a storage location where a known or unknown quantity (value) resides. This location has an associated symbolic name. In Haas machines the location is designated with a number with the pound (#) symbol in front of it.

There are three types of variables: local, global and system. Local variables are #1 thru #33. With local variables a G65, M30 or reset will clear all values to zero. Global variables have three ranges 100-199, Local and Global variables may be seen and changed by pressing the Current Commands key and using the page up or down. The title of the page will be Macro Variables. To search for a particular variable enter the variable number and search using the arrow keys. The values on these pages may be changed by highlighting the number, enter a value and press the Enter key. All the variables may be zeroed by pressing the origin key.

System variables are used by the machine tool to store particular information which the programmer has access to and may alter. They may not be altered thru the Macro Variable pages.

For example variable number #2001 stores the value for tool offset length number 1. What information a particular system variable stores may be found in the Haas Mill Operator's Manual. Some system variables are read only and may not be altered by the programmer.

G65 Macro Program Call: Macro program command **G65 Pxxxx** will call up a particular macro program. Using the G65 command allows the use of variables in the macro program called.

Probe programs use the **G31: Feed Until Skip** command. In the G31 block a specific axis move is commanded with a feed rate. When the move is started it will continue until the position is reached or the probe makes a hit. If the probe makes a hit the feed will stop and the program will skip down to the next block. An audible beep will be heard when the probe makes a hit.

9000 Programs: Probing programs are all designated as 9xxx programs. The next page gives a typical list of Renishaw programs loaded in a Haas vertical machining center. To prevent these programs from being altered **Setting 23 9xxx PROGS EDIT LOCK should be set to ON**. With setting 23 is set to ON the probe 9000 programs shown below will not viewed or accessible when the LIST PROGRAM key is depressed.

Macro Program List

<u>Macro Program Name</u>	<u>Function</u>
009023	(REN EASYSET V2.8)
009301	(REN IPS CALIBRATION V2.8)
009351	(REN IPS BORE RECT V2.8)
009352	(REN IPS BOSS BLOCK V2.8)
009353	(REN IPS WEB X V2.8)
009354	(REN IPS POCKET X V2.8)
009355	(REN IPS WEBB Y V2.8)
009356	(REN IPS POCKET Y V2.8)
009357	(REN IPS INSIDE CORNER V2.8)
009358	(REN IPS OUTSIDE CORNER V2.8)
009359	(REN IPS SINGLSE SURFACE V2.8)
009376	(REN OTS CALIBRATION)
009377	(REN OTS TOOL SET)
009378	(REN OTS TOOL SET L D)
009721	(REN X DIAM)
009722	(REN Y DIAM)
009723	(REN ACT OFFSET)
009724	(SETTINGS)
009725	(SETTINGS CHECK)
009726	(REN BASIC MEASURE -2T)
009727	(REN VECTOR OD-ID)
009730	(REN PRINT)
009731	(REN #[#161+9]\ LOAD/ATAN)
009732	(REN OFFSET TYPE)
009799	(REN SETTING)
009801	(REN CAL PROBE LENGTH)
009802	(REN CAL XY OFFSET)
009803	(REN CAL STYLUS RAD)
009804	(REN CAL VEC STYLUS RAD)
009810	(REN PROTECTED POSN)
009811	(REN XYZ MEASURE)
009812	(REN WEB POCKET MEASURE)
009814	(REN BORE BOSS MEASURE)
009815	(REN INTERNAL CORNER)
009816	(REN EXTERNAL CORNER)
009817	(REN 4 TH AXIS X MEASURE)
009818	(REN 4 TH AXIS Y MEASURE)
009819	(REN PCD BORE BOSS)
009820	(REN STOCK ALLOWANCE)
009821	(REN ANG XY SINGLE SURFACE)
009822	(REN ANG WEB POCKET)
009823	(REN 3-POINT BORE BOSS)
009830	(REN MULTI-STYLUS STORE)
009831	(REN MULTI-STYLUS LOAD)

<u>Macro Program Name</u>	<u>Function</u>
O09832	(REN OMP 40 ON)
O09833	(REN OMP 40 OFF)
O09834	(REN FEATURE TO FEATURE)
O09835	(SPC OFFSET UPDATE)
O09843	(REN XY PLANE ANGLE)
O09850	(RENISHAW TOOL SELECT)
O09851	(RENISHAW LENGTH SET)
O09852	(RENISHAW DIA. SET)
O09853	(RENISHAW AUTO SET)
O09854	(RENISHAW DELAY)
O09855	(REN OTS ON)
O09856	(REN OTS OFF)
O09995	(WIPS FOR RENISHAW V2.8)
O09996	(HAAS VQC WIRELESS PROBE Eng Inc)

From the list above some 60 programs are used with different Renishaw probing applications. With **WIPS** (Wireless Intuitive Probe System) and **VQC** (Visual Quick Code) the appropriate programs are selected and variable information like diameter, work offset, or tool number are inserted into the different probe programs. Then work offset or tool length information is inserted into the Haas controls after the programs are run. This greatly simplifies using probes to get information like work offsets and tool length offsets. Probe selection is further simplified with visual screens showing the various options for getting tool lengths for different tools. Also the various features that may be used to determine work offsets are easy to pick out from the different screens available.

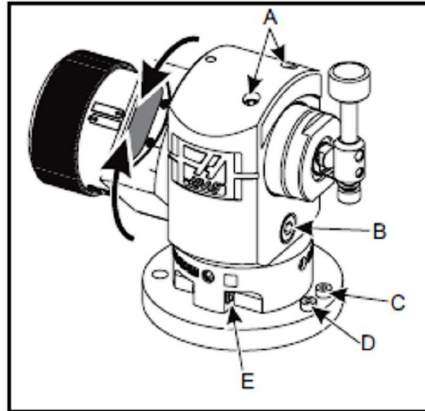
Visual Quick Code(VQC) was developed by Haas in conjunction with Renishaw probes and programs. It was developed so that using probes to set up Haas machining centers was relatively simple and easy to use. **VQS will only work with a Renishaw Probing System.**

Wireless Intuitive Probing System (WIPS) is the latest probing software developed by Haas. It may be used for either Renishaw or Marposs probes.

Probe Alignment

The OMI, OTS and the OMP must be aligned so that the infrared light will be able to be detected by the photodiodes from the machine interface (OMI) to both the spindle probe (OMP) and table OTS and vice versa. Assuming the probes are able to communicate with each other (see Haas ES0583 Haas Quick Start and Set Up) for recommended alignment locations.

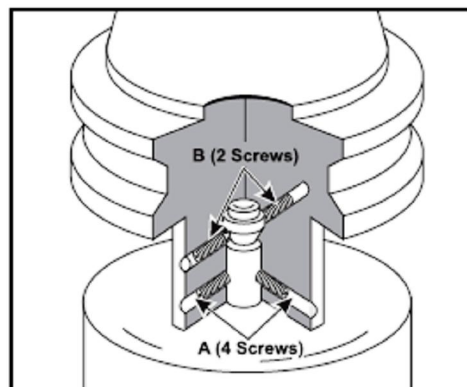
For accuracy the **table probe (OTS)** must be within .0001 inch parallel with the machine X and Y axis plane. Place a .0001 test indicator on the spindle with an Indicol or other method. Move indicator so it touches the top surface of the probe. The top surface of the probe cylindrical stylus should be within .001 inch when the table is moved in both the X and Y directions.



For **adjustments in the X axis** first loosen **screw C** above. Then **screw D** which acts like a jack screw can be adjusted so there is less than .001 inch change from one side to the other in the X axis. When properly adjusted tighten screw C down.

When moving the table in the **Y axis, adjustment** may be made by alternately adjusting the two **A screws**.

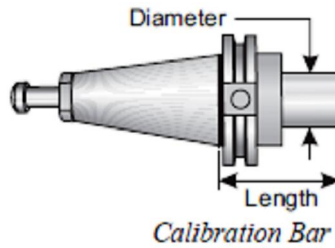
For the **spindle probe (OMP)** alignment place the indicator on the stylus ruby ball. Rotate the spindle. If run out of more than .0002 inch is detected, adjust the spindle probe by first loosening the two B screws. The four A screws may then loosened and tightened until there is run out below .0002". Then tighten down the B screws. See below:



VQCPS – Visual Quick Code Probe System

Probe Calibration

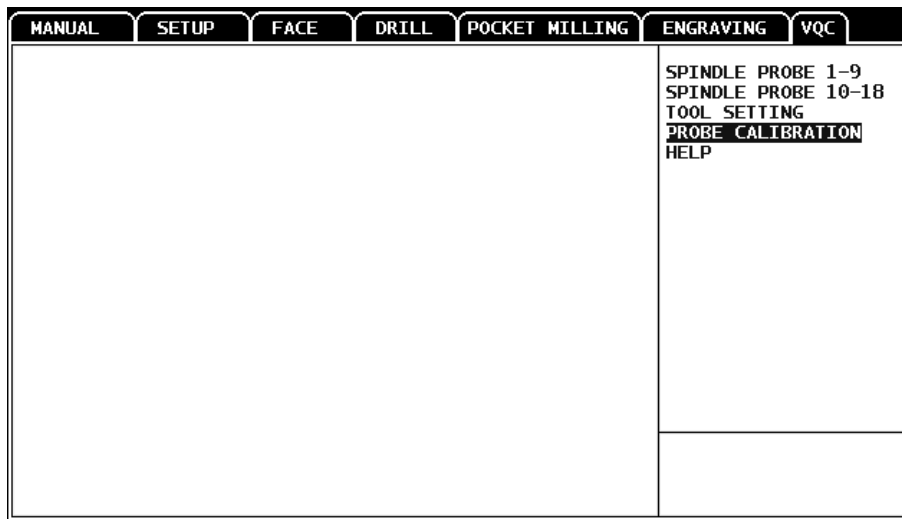
To calibrate the probe **first** a standard must be purchased or created. The calibration bar as pictured below is simply a ground pin of known diameter placed in a tool holder. The other information which is needed is the length from the end of the tool to the gauge line of the tool. This gauge length may be determined using a Parlec Tool Setting machine.



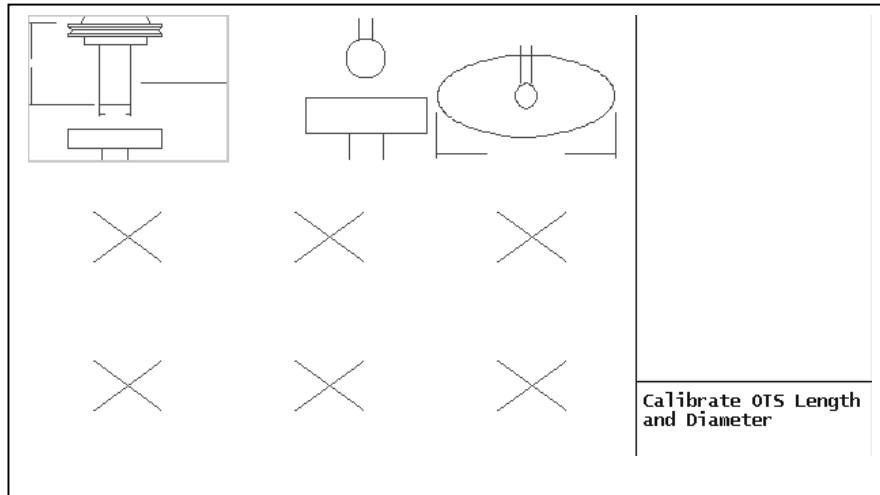
The second tool needed is a calibrated ring gage. This must be clamped to the table.

After the spindle and table probes are aligned for run out and parallelism respectively Visual Quick Code (VQC) must be accessed thru the Haas control.

To access VQC press the **MDI** key and then the **Prgm Convrs** key to open the IPS tabulated screen. Cursor to the **VQC** tab at the far right and press enter to view screen below. If the VQC screen below does not come up press the F2 and highlight program 009996 (HAAS VQS WIRELESS PROBE Eng Inc) then press enter. Highlight the PROBE CALIBRATION line as seen below and press enter.

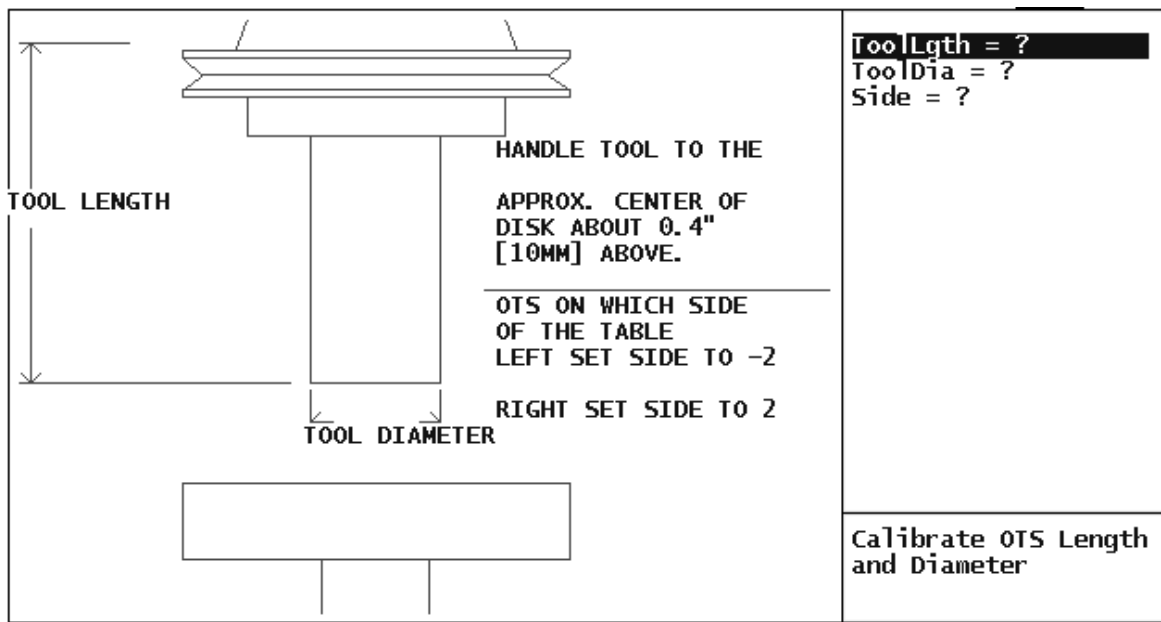


This will bring you to the following screen. Highlight the picture in the upper left hand corner of the screen below (**Calibrate the OTS length and Diameter**) then press Enter.

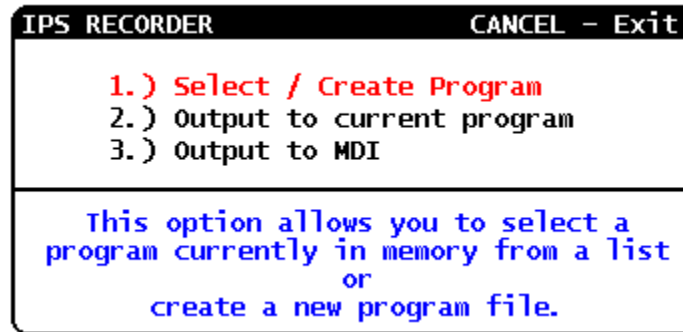


The following screen will appear below. The first thing that must be done is jog the bar standard to the center of the table probe disk and about 0.4" above it. The machine is already in the **IPS Jog Mode**. The machine will not have to put if Set Up Jog mode. If you put it in the Set Up Jog handle mode you will exit the VQC screen.

From the screen below enter the Tool Length and Diameter of the Bar Standard. Also the side that the table probe is on must be indicated by either a -2 or 2. Normally the table probe is located in the right hand side of the table so a 2 is entered.



After entering the side the table probe is on the following pop up will appear. Cursor to option #3 (Output to MDI) and press enter.



This will put the calibration program in MDI shown below.

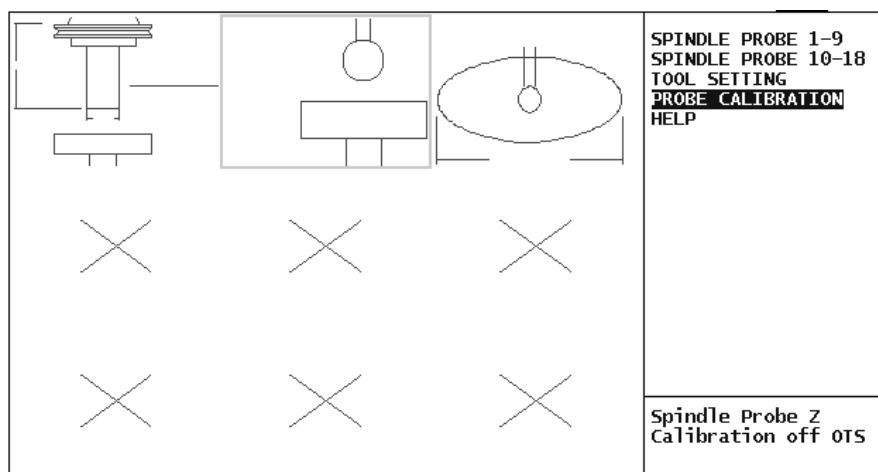
```

;
(Calibrate OTS Length and Diameter) ;
(ToolLgtb = 5.1) ;
(ToolDia = 0.5) ;
(Side = 2.) ;
;
G00 G90 ;
G65 P9023 A20. K5.1 S0.5 D2. ;
;

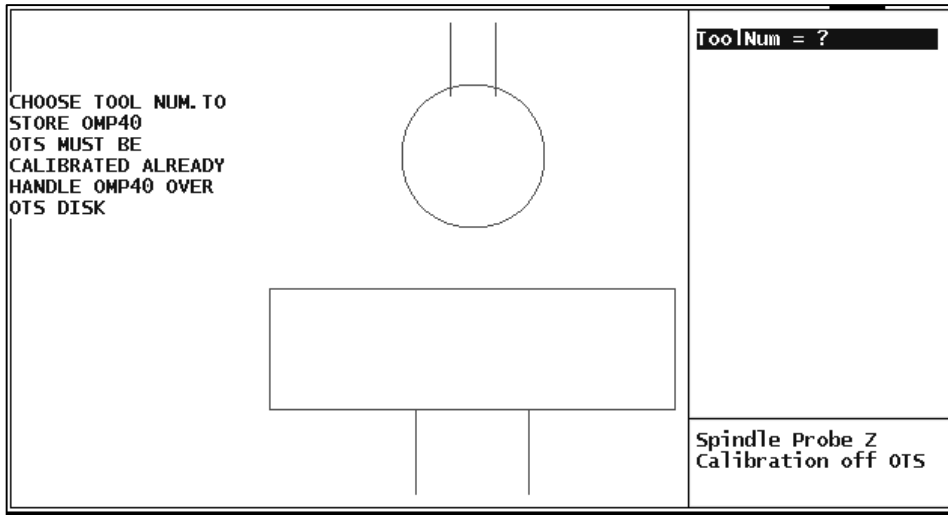
```

Run this program in MDI. The Diameter and the X,Y,Z location of the table probe disk will be stored in the machine controls to be used later. It uses this information to determine the Tool Length and Diameter when setting tools.

The next step is to calibrate the Z of the spindle probe. Index the machine to the tool where the spindle probe is located. Usually this is designated as the last tool. In the Tool Pot Location screen the spindle probe is always designated as a **heavy tool**. Because of the delicate nature of the electronics in the spindle probe, rapid indexing may disturb the calibration of the spindle probe especially in the SS machines. Next get back in the IPS Screen by pressing the **Prgm Convrs** key, open the VQC tab and select the Spindle Probe Z Calibration seen below.



The following screen will appear. Move the spindle probe just .400" above the table probe disk.



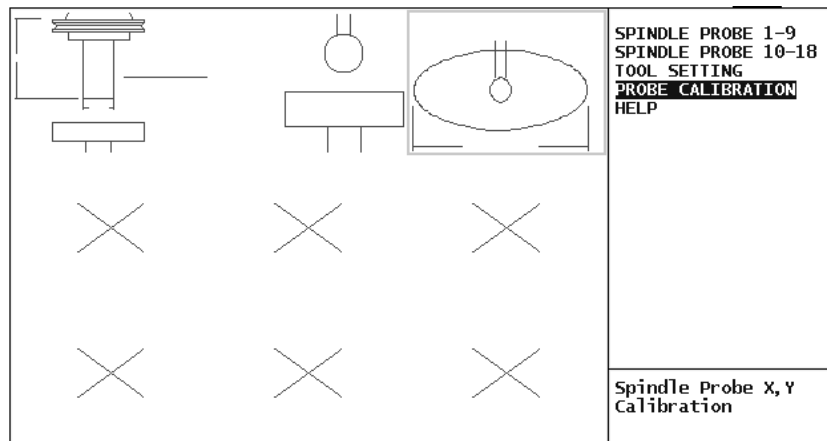
Entering the Tool Number will bring up the IPS Recorder Screen. Select number three and following code will appear in MDI program window.

```

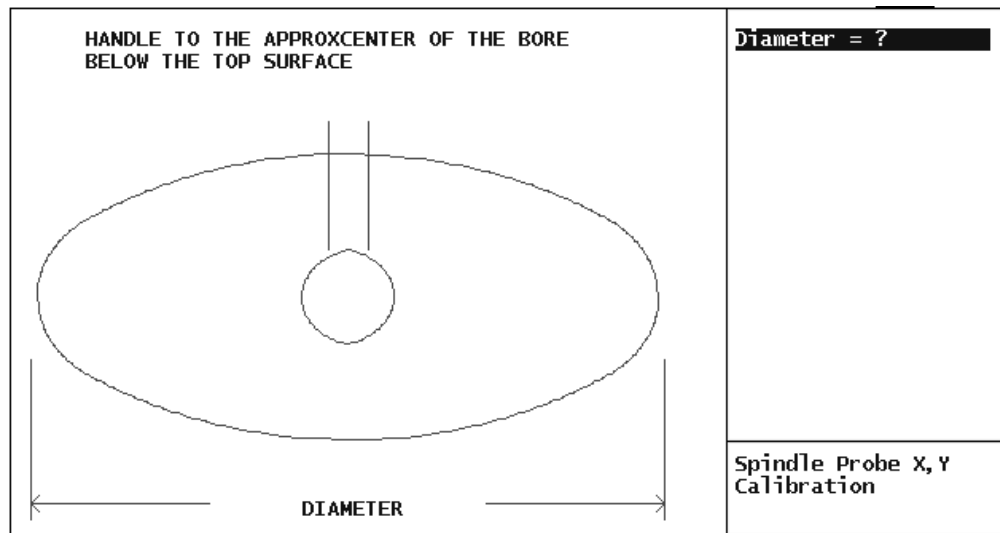
;
;(Spindle Probe Z Calibration off OTS) ;
;(ToolNum = 25) ;
;
G00 G90 ;
G65 P9023 A21. T25 ;
;
;
;

```

For the final step in calibrating the spindle probe, the diameter of the ruby ball and location in X and Y needs to be determined. Open up the VQC screen and select (Spindle Probe X,Y Calibration). See below screen shot.



Selecting the third option will give the screen below.



As the text explains handle the spindle probe to the approximate center of the ring gage. Then move the probe down in Z till it is .3 to.4" below the top face of the ring gage. Entering the known diameter of the ring gage will bring up the recorder screen. Output the code to MDI. The following code will appear in the MDI screen.

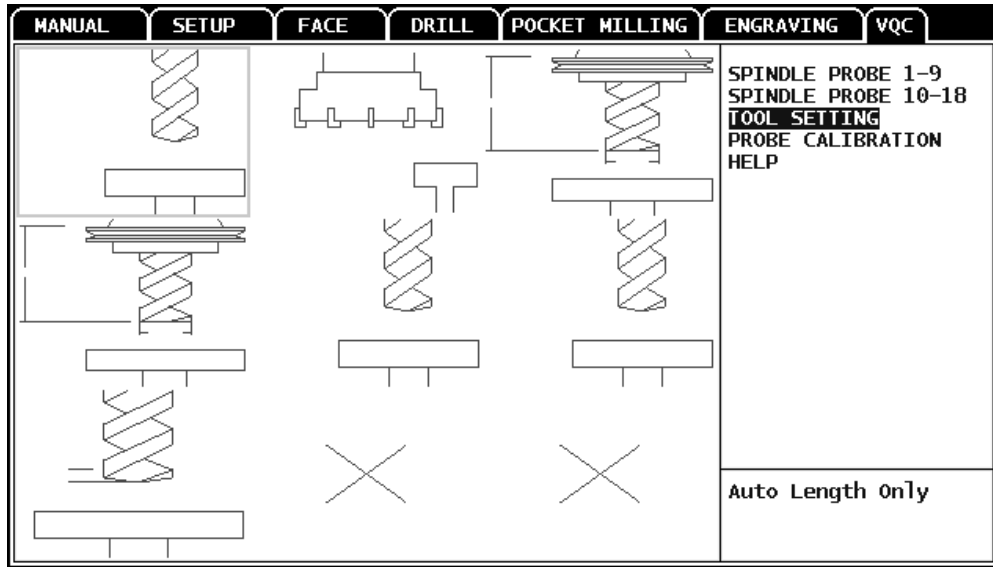
```
;  
(Spindle Probe X,Y Calibration) ;  
(Diameter = 1.) ;  
;  
G00 G90 ;  
G65 P9023 A10. D1. ;  
;  
-
```

Run the above code in MDI mode and the WIPS calibration is **complete**.

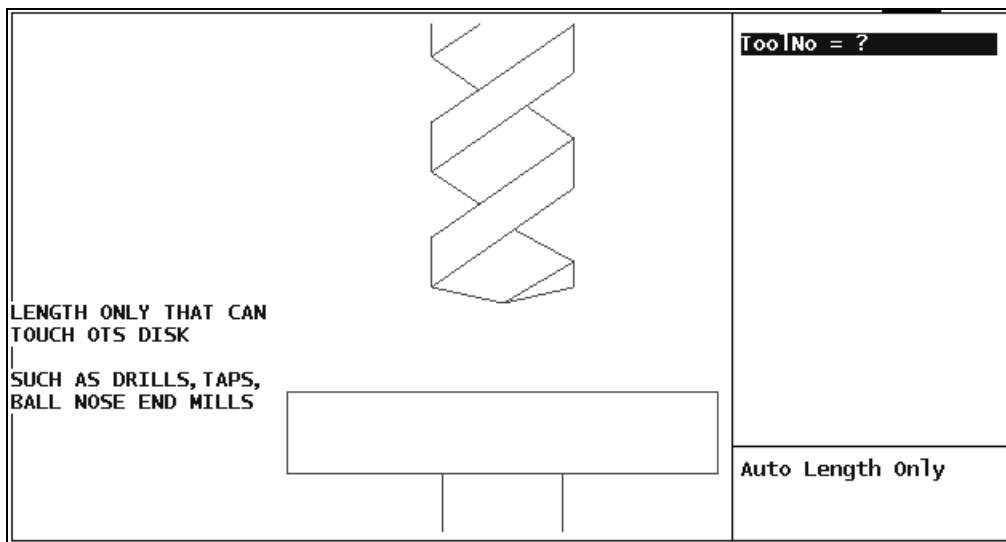
Setting of Tool Length Offsets using VCQ

Drill Type Tools:

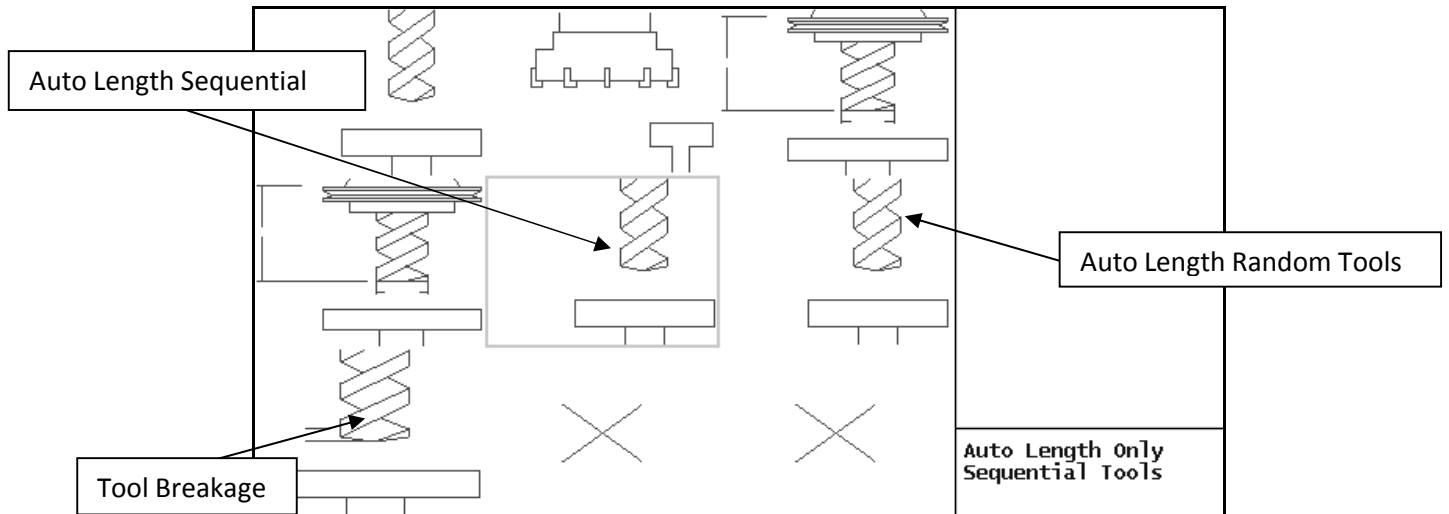
Open VQC and highlight the Tool Setting. Pressing the Enter Key will open VQC tool setting options. See below:



The VQC first option is **auto length only**. This is used for drills, taps, ball nose end mills, and reamers. These tools only require information on the tool gauge length to work properly. Selecting the first option will give the following screen:



Enter the tool number and the Recorder Screen appears. Put the code in MDI. The machine will index to the tool entered, rapid to the center of the OTS disk, slowly move down until a hit is made, back off and take another hit and rapid back to home in Z.

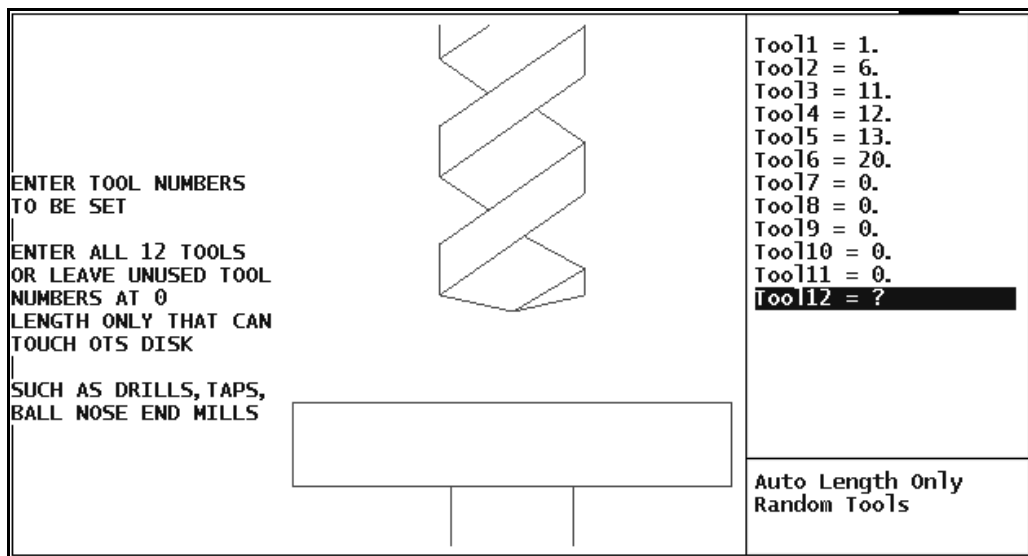


VQC gives two other options using the auto length option. Several drill type tools in sequence may be probed using VCQ (see **Auto Length Sequential Tools** above).

```
Frsttool = 1.  
Lasttool = ?
```

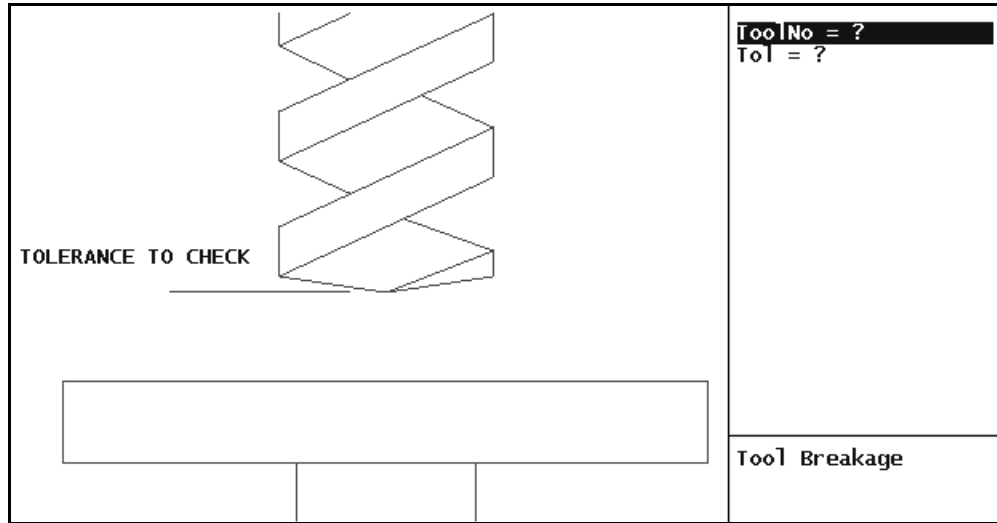
List the first tool and last tool above. VQC will probe the first, the last tool and all the tools in between.

Selecting **Auto Length Random Tools** gives the following:



Enter drill type tools that need probing, put zero in the rest by pressing Enter key. Only the tools indicated will be probed.

Selecting **Tool Breakage** gives the following screen.

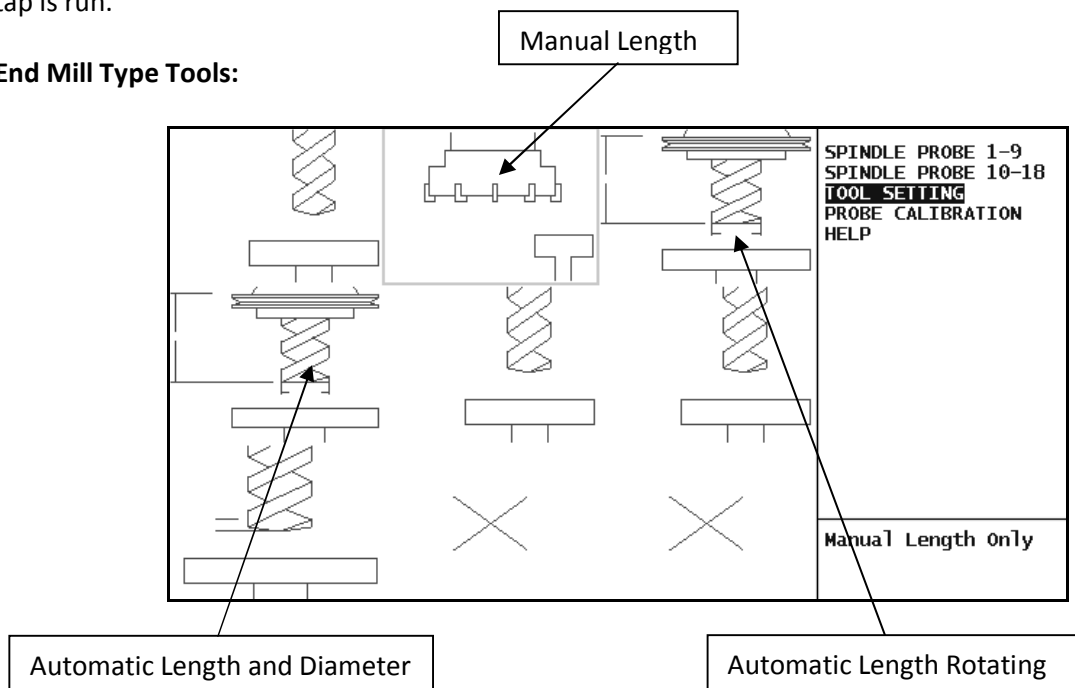


This option will check the tool for any changes in length outside of a tolerance given. If the Tool Length Offset is outside of the tolerance given the alarm code 1099 Broken Tool appears and stops the machine. See following code:

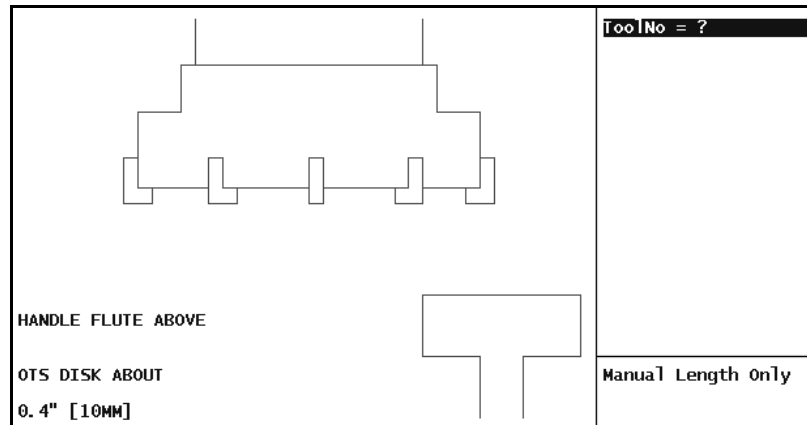
```
(Tool Breakage) ;
(ToolNo = 10) ;
(Tol = 0.03) ;
G28 G91 Z0 ;
G90 G49 ;
T10 M06 ;
G00 G90 ;
G65 P9023 A24. T10 H0.03 ;
```

This code may be inserted into a regular program using the IPS recorder to check if a tool has broken. This is used when a machine is run unattended for example in lights out overnight. To stop the machine from crashing all the rest of the tools in a program a tap drill is checked using the code above before a tap is run.

End Mill Type Tools:

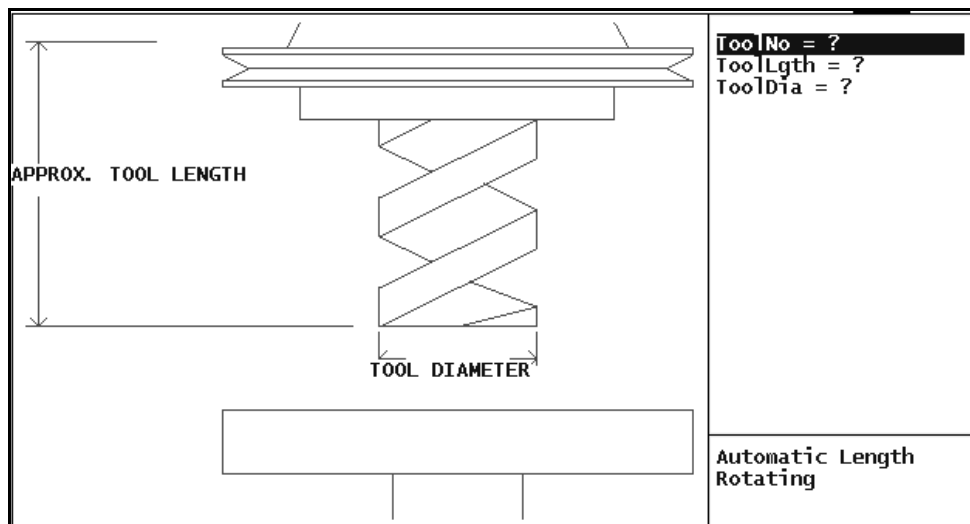


There are three ways End Mill Type tools maybe checked. The first way is a Manual Length check which may be used to check a Face Mill where only the tool length of the tool needs to be measured. Selecting Manual Length opens the following window.



With this option an insert of a face mill would be jogged to the center of the table probe disk. The program run in MDI and the tool length noted. The tool may next be rotated to the next insert on the cutter and the program in MDI run again and so on until all the inserts in the cutter have been checked. The largest Tool Length Offset would be used as this is this insert determines how much material is cut off the face of the part.

The **Automatic Length Rotating** option is also used for determined just the length of an end mill type tool.



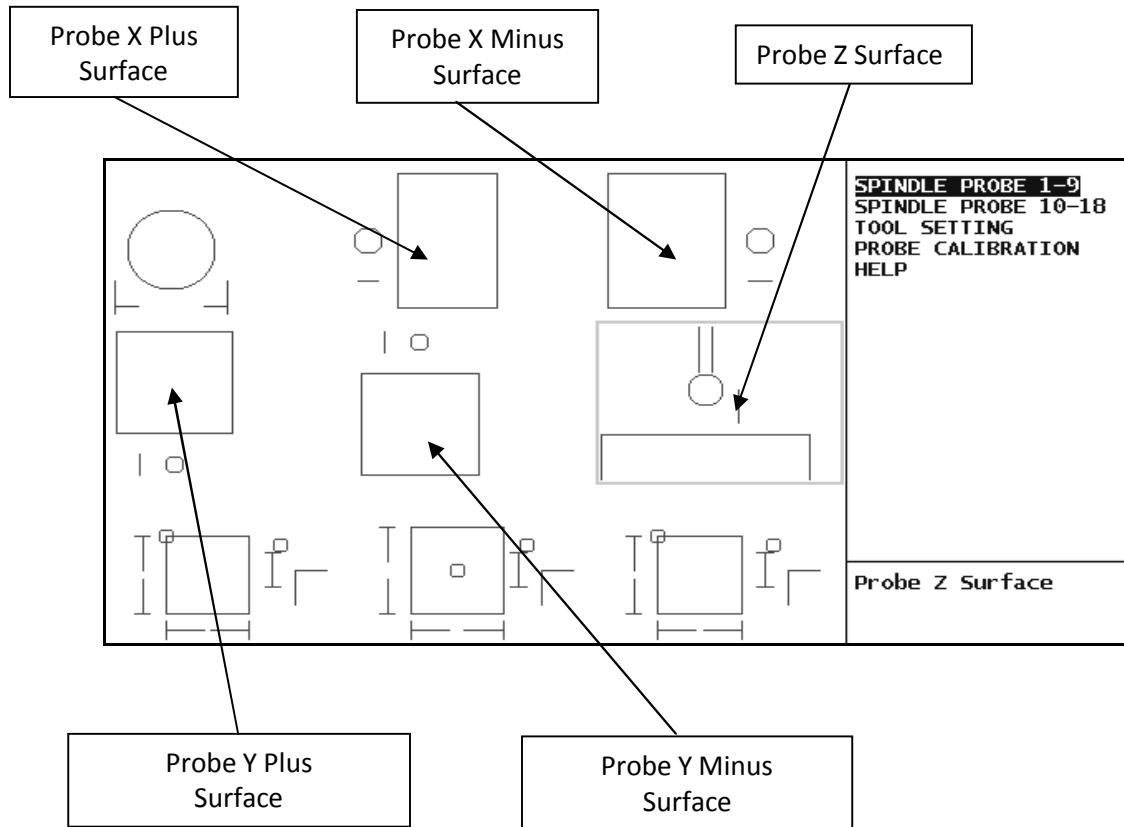
The tool length measurement with a scale must be entered along with the tool diameter. The program created will bring the tool down to about 2", offset the tool to the left over a certain diameter so that the flutes will hit the center of the disk and start rotating. The tool will then come down slowly and take a couple of hits and return to Z home position. **End mills must use this method or mismatch** between cutters will take place.

The **Automatic Length and Diameter** option needs the same information entered and works basically the same as the Automatic Length. The only thing different before the tool goes home the end mill will move to the front and move down in Z, move back taking a hit, move up over the top of the disk and down and take a rotating hit on the backside. This determines the diameter of the tool probed.

Setting Work Offsets using Visual Quick Code

Manually probing single surfaces to determine X, Y, or Z for work offsets:

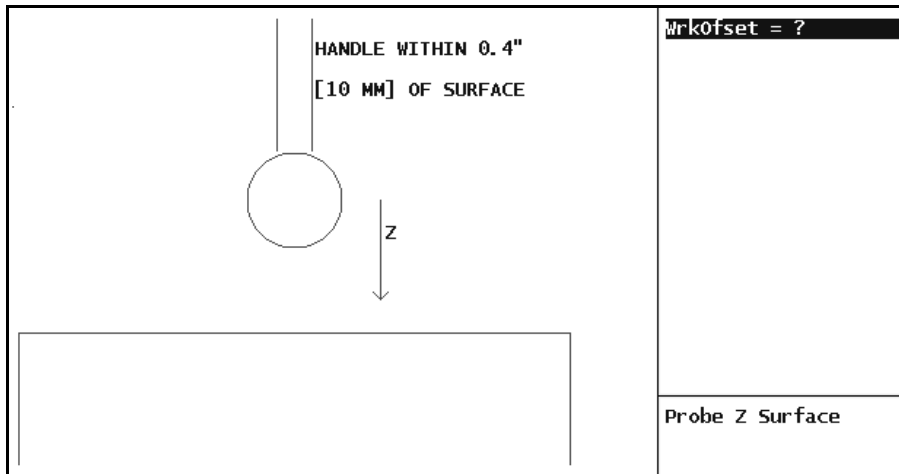
Entering Visual Quick Code and selecting (**SPINDLE PROBE 1-9**) gives the following screen:



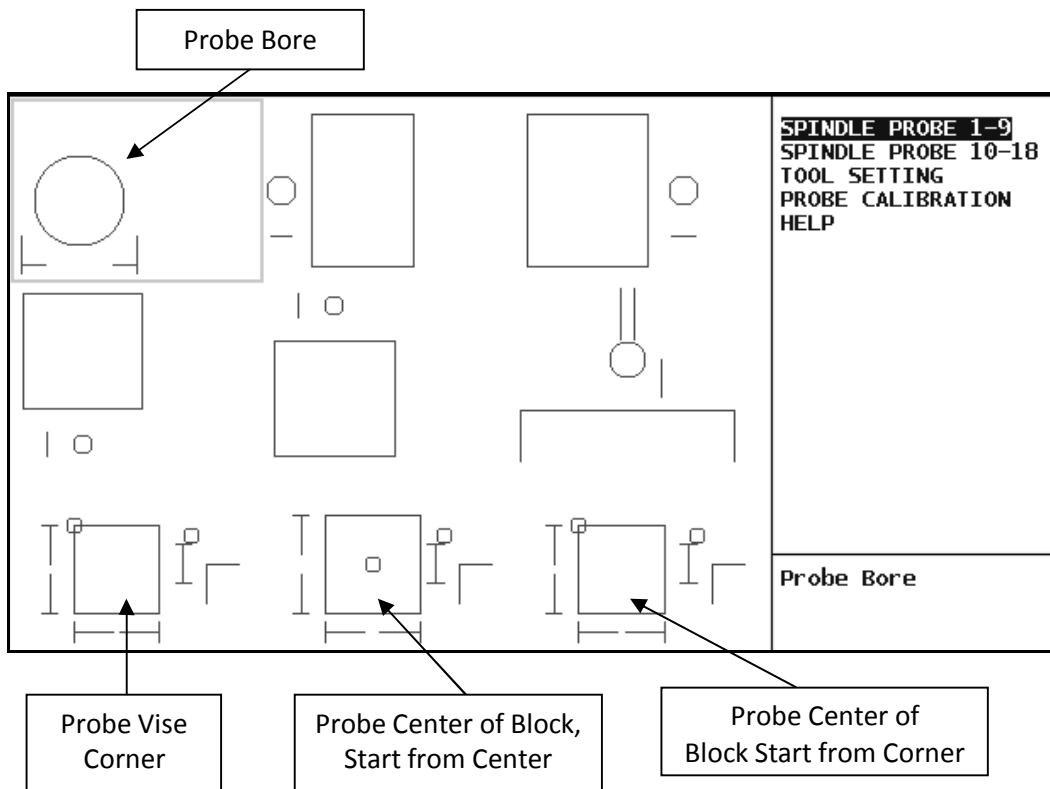
All the above are manual probing of single surfaces. Select the surface to be probed and the plus or minus direction the probe will be moving. Basically they work the same, see the next page for probing the Z axis direction.

As with all the above selections the probe is handle to within 0.4" from the surface that is being probed. Then the Work Offset number is entered: **54 for G54 or 154.01 for G154 P1**. The code is put into MDI thru the IPS Recorder. Then the probing program is run in MDI and the respective X,Y, or Z surface value for the work offset is placed in the work offset page.

Screen for Probing Z surface



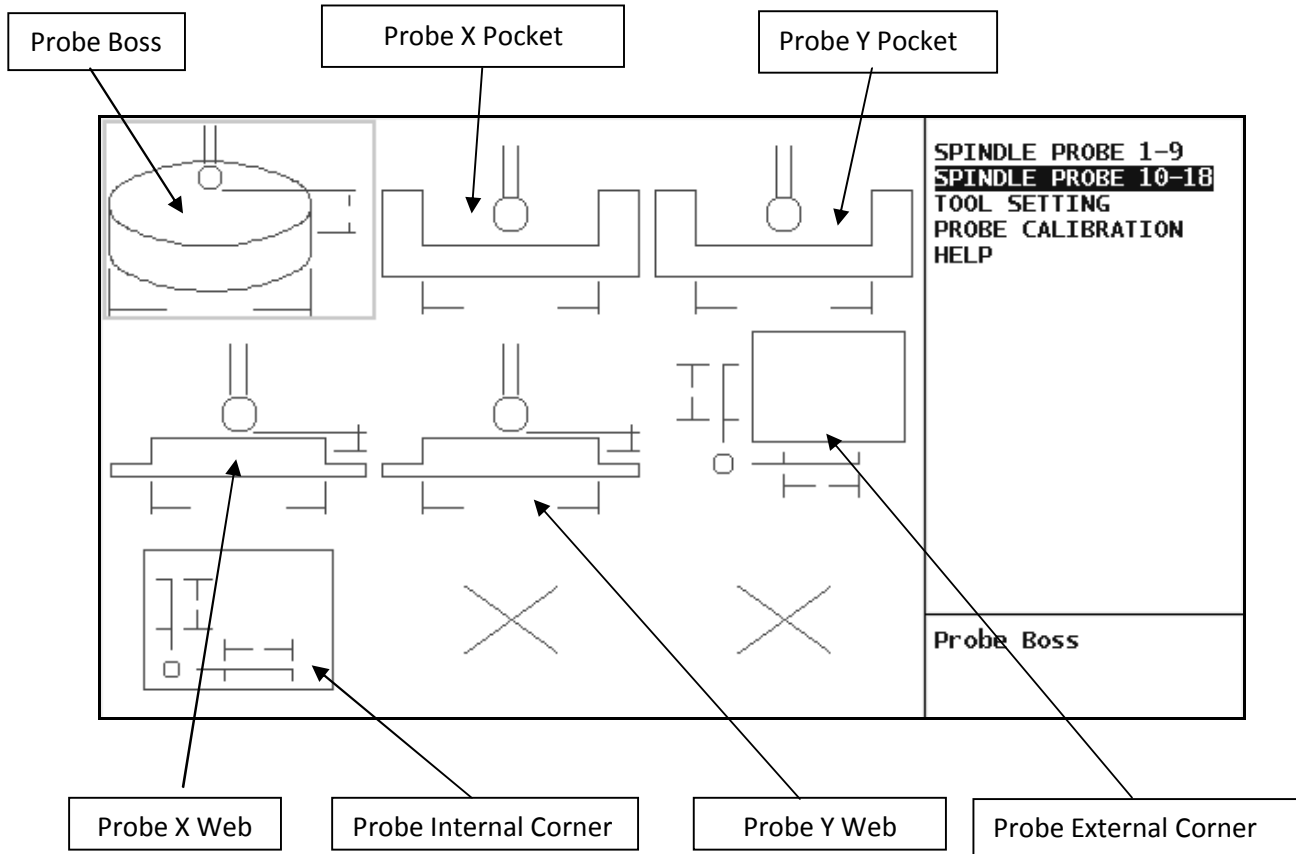
Other Options for using probes to determine Work Offsets



The above options are self explanatory. Using the Probe Bore function will place the probe when it is done in the center of the bore. The diameter of the bore is located in variable #188.

Probe Vise Corner is commonly used. Jog probe directly over the center of the upper left hand corner. Enter a **negative value** for the Z, usually -.4. Give the length of rough stock in X and Y. The probing program is run MDI taking hits in X, Y, and Z. Values will be put on the work offset for X, Y, and Z.

Spindle Probe 10-18 Other Options for Work Offsets



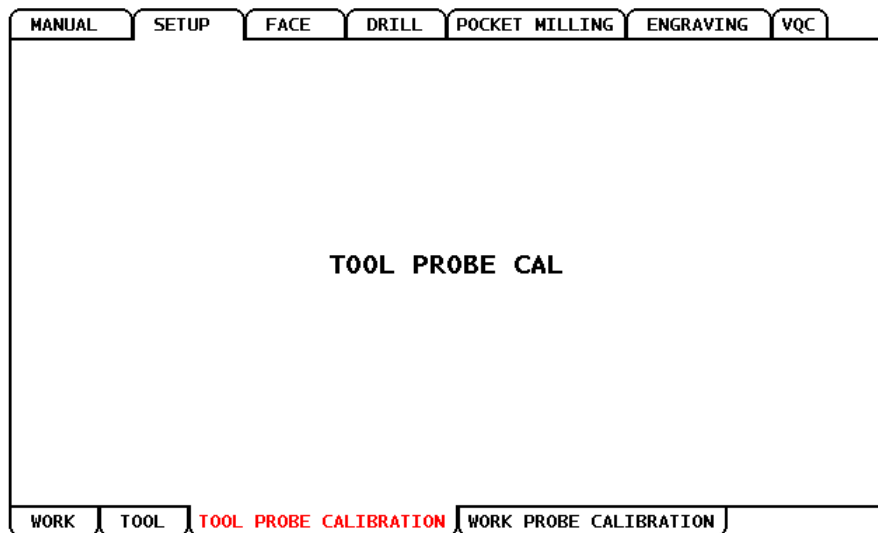
Uses of the above routines are self explanatory. They give additional options for special applications.

WIPS – Wireless Intuitive Probe System

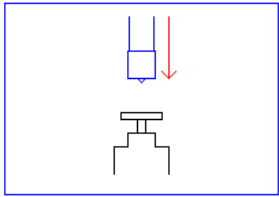
Probe Calibration

Probe calibration using VQC and IPS are very similar using almost all the same macro programs. The difference is the how the tool length of the spindle probe is determined. In the VQC the spindle probe tool is determined by probing the disk on the calibrated table probe. In IPS the tool length of the spindle probe is calibrated by probing the top of the ring gage whose position in Z has been determined by touching off the top with the calibration bar.

To enter the calibration mode in IPS enter the IPS tabulated menu by pressing MDI and then PRGM CONVRS keys. On the top cursor to the set up tab and press enter to get to the lower tabs. Highlight **Tool Probe Calibration** tab as below.



Pressing Enter key will bring up the following screen:

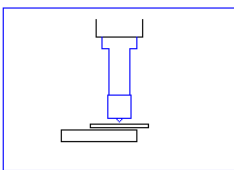
MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
Z POSITION	0.0000 in	TOOL OFFSET	1			
X POSITION	0.0000 in	LENGTH	0.0000 in			
Y POSITION	0.0000 in	DIAMETER	0.0000 in			
CALIBRATION INCOMPLETE						
WORK	TOOL	TOOL PROBE CALIBRATION	WORK PROBE CALIBRATION			

Jog the Z-axis down until just above the table probe. Press F1 to record this position.

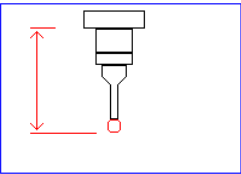
Press **CANCEL** to exit current mode

Place the calibration bar in the spindle. Then handle the bar to about .3" above the table probe disk and approximately on the center. Follow the instructions on the bottom of the page, press **F1**. After storing the machine coordinates the X Position box will be highlighted in red. Press **F1** again. Then Y position box will be highlighted. Press **F1** again. Now follow the instructions on the bottom by first entering the tool number of the calibration bar, then enter the tool length, then the Diameter. Then press Cycle Start and the calibration program will be run. After it is successfully run then the calibration box will indicate that it is **COMPLETED**.

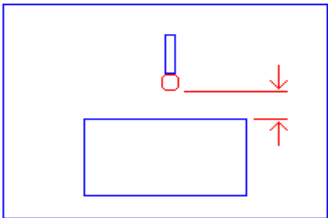
Close the tool probe calibration tab, cursor to the **Work Probe Calibration** tab and open by pressing Enter key. The following screen will appear. Follow instructions by bringing calibration bar over calibration ring and touching off with a shim. As instructions indicate press **F1**. Enter the known gage length of the calibration bar, then the thickness of the shim.

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
Z POSITION	0.0000 in	PROBE LENGTH	0.0000 in			
TOOL LENGTH	0.0000 in	BALL DIAMETER	0.0000 in			
SHIM	0.0000 in	CAL MASTER DI	0.0000 in			
CALIBRATION INCOMPLETE						
WORK	TOOL	TOOL PROBE CALIBRATION	WORK PROBE CALIBRATION			
<p>Warning: Change tool to tool of known length before continuing.</p>				<p>Place a shim on the Z surface of the cal. ring. Jog the Z-axis down until you touch the shim. Press F1. Press CANCEL to exit current mode</p>		

After the shim's width is entered the machine **must be indexed to the spindle probe** as indicated by the bottom of the screen below.

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
Z POSITION -0.5127 in	PROBE LENGTH 0.0000 in					
TOOL LENGTH 5.2100 in	BALL DIAMETER 0.0000 in					
SHIM 0.0000 in	CAL MASTER DI 0.0000 in					
CALIBRATION INCOMPLETE						
WORK	TOOL	TOOL PROBE CALIBRATION	WORK PROBE CALIBRATION			
<p>Warning: Change tool to work probe before continuing.</p>			<p>Enter the approximate length of the work probe that is being calibrated. Press ENTER to record this value. Press CANCEL to exit current mode</p>			

Measure the spindle probe tool length with a scale and enter the value in the probe length box. Measure the ruby ball diameter at the end of the probe and enter the value in the ball diameter box. The standard probe diameter is 10.0 mm or .3935". Next enter inside diameter of the calibration ring gage. The following screen should appear.

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
Z POSITION -0.5127 in	PROBE LENGTH 5.1000 in					
TOOL LENGTH 5.2100 in	BALL DIAMETER 0.3935 in					
SHIM 0.0040 in	CAL MASTER DI 1.0000 in					
CALIBRATION INCOMPLETE						
WORK	TOOL	TOOL PROBE CALIBRATION	WORK PROBE CALIBRATION			

Next follow directions on the bottom of screen, center the probe .3' above the ring gage and press CYCLE START

Position probe 0.3 inches above the center of Calibration Master. Press CYCLE START to begin calibration. Press CANCEL to exit current mode

After running the calibration program successfully the **INCOMPLETE** text in the calibration box will turn to **COMPLETED**.

Setting of Tool Length Offsets using IPS

IPS uses the same programs to set tools as VQC. The only difference is the navigation of the controls. With VQC the VQC system is entered thru the IPS screen by opening the VQC tab. With IPS tool length probing and work offset probing is entered thru the offsets pages. To enter the IPS probe programs the machine must be either in the MDI mode or the EDIT mode. IPS tool and work offset setting may not be accessed in the MEMORY Mode.

In the MDI mode the following TOOL OFFSET screen may be accessed by pressing the Offset key. If the WORK OFFSET page comes up press the Offset key again and the following screen will appear. Note at the top of screen the << PROBING and the TOOL INFO >> They give the direction using the cursor. These will not appear in the Memory Mode.

<< PROBING		TOOL OFFSET				TOOL INFO >>	
IPS ON	COOLANT	H(LENGTH)		D(DIA)			
TOOL	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR		
1 SPINDLE	0	0.	0.	0.	0.		
2	0	7.0004	0.	0.2109	0.		
3	0	7.0004	0.	0.2500	0.		
4	0	5.0000	0.	0.0890	0.		
5	0	5.0000	0.	0.0910	0.		
6	0	5.0000	0.	0.1250	0.		
7	0	5.0000	0.	0.	0.		
8	0	5.0000	0.	0.	0.		
9	0	-10.0000	0.	0.	0.		

Using the right arrow cursor key will highlight the Tool Type Column

<< TOOL OFFSET		TOOL INFO		PROBING	
IPS ON	FLUTES	ACTUAL DIAMETER	TOOL TYPE		
1 SPINDLE	5	0.	0-NONE		
2	2	0.	1-DRILL		
3	2	0.	2-TAP		
4	2	0.	1-DRILL		
5	2	0.	2-TAP		
6	2	0.	4-END MILL		
7	2	0.	0-NONE		
8	2	0.	0-NONE		
9	2	0.	0-NONE		

0-NONE 1-DRILL 2-TAP 3-SHELL MILL 4-END MILL 5-SPOT DRILL 6-BALL NOSE 7-PROBE



No tool type has been selected. Pressing the 3 key and pressing the ENTER key will designate Tool #1 a Shell Mill. See next page.

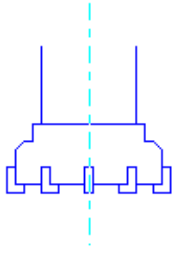
<< TOOL OFFSET		TOOL INFO		PROBING
IPS ON		ACTUAL	TOOL	
TOOL	FLUTES	DIAMETER	TYPE	
1 SPINDLE	5	0.	3-SHELL MILL	
2	2	0.	1-DRILL	
3	2	0.	2-TAP	
4	2	0.	1-DRILL	
5	2	0.	2-TAP	
6	2	0.	4-END MILL	
7	2	0.	0-NONE	
8	2	0.	0-NONE	
9	2	0.	0-NONE	

0-NONE 1-DRILL 2-TAP 3-SHELL MILL 4-END MILL 5-SPOT DRILL 6-BALL NOSE 7-PROBE

Next the type of Tool Probing is selected. Right cursor to the PROB /TYPE column. Four different options are given.

<< TOOL INFO		PROBING			TOOL OFFSET >>
IPS ON	APPROX	APPROX	EDGE	TOOL	PROB
TOOL	LENGTH	DIAMETER	HEIG	TOLERANC	TYPE
1 SPINDLE	0.	0.	0.	0.	3-LEN & DIA
2	0.	0.	0.	0.	0-NONE
3	0.	0.	0.	0.	0-NONE
4	0.	0.	0.	0.	0-NONE
5	0.	0.	0.	0.	0-NONE
6	0.	0.	0.	0.	0-NONE
7	0.	0.	0.	0.	0-NONE
8	0.	0.	0.	0.	0-NONE
9	0.	0.	0.	0.	0-NONE

0-NONE 1-LENGTH (ROTATING) 2-LENGTH (NON-ROTATING) 3-LEN & DIA (ROTATING)

TOOL PROBE HELP	Tool Type: SHELL MILL
<p>Selected the type of probing to be performed:</p> <ul style="list-style-type: none"> 0 - No tool probing to be performed. 1 - Length probing (Rotating). 2 - Length probing (Non-Rotating). 3 - Length and Diameter probing (Rotating). <p>Press the [TOOL OFFSET MEASUR] key to start the Automatic Probing Options.</p>	

Enter a **zero** and **no probing** will take place. **Option 2** is used for **drill type tools**. **Option 1** is a rotating probe for just getting the **tool length** for tools like **end mills**. **Option 3** is used for getting **tool length** and **diameters** for **end mill type tools**.

If you press **Tool Offset Measur** key and then cycle start to probe the tool a message will come up noting that the tool does not have all its inputs.

Cursor to the left to give the approximate length column will give the following screen. In the lower right hand corner is a picture describing the information is needed. Note that even though a value has been entered into the approximate length column the **Prob Type Column** is highlighted in red. This denotes it does not have all the information to run the probe program.

<< TOOL INFO		PROBING			TOOL OFFSET >>	
IPS ON	APPROX	APPROX	EDGE	TOOL	PROB	
TOOL	LENGTH	DIAMETER	HEIG	TOLERANC	TYPE	
1 SPINDLE	3.0000	0.	0.	0.	3-LEN & DIA	
2	0.	0.	0.	0.	0-NONE	
3	0.	0.	0.	0.	0-NONE	
4	0.	0.	0.	0.	0-NONE	
5	0.	0.	0.	0.	0-NONE	
6	0.	0.	0.	0.	0-NONE	
7	0.	0.	0.	0.	0-NONE	
8	0.	0.	0.	0.	0-NONE	
9	0.	0.	0.	0.	0-NONE	

ENTER A VALUE. PRESS [WRITE] TO ADD OR [F1] TO SET THE VALUE.

TOOL PROBE HELP	Tool Type: SHELL MILL
<p>Enter the approximate length of the tool to be measured.</p> <p>(Enter a positive number only).</p>	
<p>Press the [TOOL OFFSET MEASUR] key to start the Automatic Probing Options.</p>	

Entering information on diameter and how far down the tool will go to get diameter information gives the following screen. Note all the information that is needed has been entered and the Prob/Type column has turned green. Now the tool probe program may be run.

<< TOOL INFO		PROBING			TOOL OFFSET >>	
IPS ON	APPROX	APPROX	EDGE	TOOL	PROB	
TOOL	LENGTH	DIAMETER	HEIG	TOLERANC	TYPE	
1 SPINDLE	3.0000	2.0000	0.2000	0.	3-LEN & DIA	
2	0.	0.	0.	0.	0-NONE	
3	0.	0.	0.	0.	0-NONE	
4	0.	0.	0.	0.	0-NONE	
5	0.	0.	0.	0.	0-NONE	
6	0.	0.	0.	0.	0-NONE	
7	0.	0.	0.	0.	0-NONE	
8	0.	0.	0.	0.	0-NONE	
9	0.	0.	0.	0.	0-NONE	

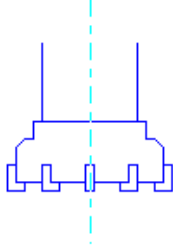
ENTER A VALUE. PRESS [WRITE] TO ADD OR [F1] TO SET THE VALUE.

TOOL PROBE HELP	Tool Type: SHELL MILL
<p>Enter the distance from where the diameter is to be measured to the bottom of the tool.</p> <p>(Enter a positive number only).</p>	
<p>Press the [TOOL OFFSET MEASUR] key to start the Automatic Probing Options.</p>	

When the Prob/Type turns green pressing [TOOL OFFSET MEASUR] key. The following Automatic Probe Options menu comes up.

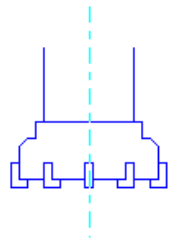
<< TOOL INFO		PROBING			TOOL OFFSET >>	
IPS ON	APPROX	APPROX	EDGE	TOOL	PROB	
TOOL	LENGTH	DIAMETER	HEIG	TOLERANC	TYPE	
1 SPINDLE	3.0000	2.0000	0.2000	0.	3-LEN & DIA	
2	0.	0.	0.	0.	0-NONE	
3	0.	0.	0.	0.	0-NONE	
4	0.	0.	0.	0.	0-NONE	
5	0.	0.	0.	0.	0-NONE	
6	0.	0.	0.	0.	0-NONE	
7	0.	0.	0.	0.	0-NONE	
8	0.	0.	0.	0.	0-NONE	
9	0.	0.	0.	0.	0-NONE	

ENTER A VALUE. PRESS [WRITE] TO ADD OR [F1] TO SET THE VALUE.

AUTOMATIC PROBE OPTIONS	Tool Type: SHELL MILL
<ul style="list-style-type: none"> * Probe selected tool. * Probe selected tool manually. * Probe selected tool for breakage/wear. * Probe all tools. * Cancel probe selection. 	
Press [CYCLE START] to run probe or "[INSERT] to copy probe code to the clipboard.	

The **first option** highlighted in yellow is to **(Probe selected tool)**. To probe the Tool 1 to get tool length and diameter press [CYCLE START]. The machine will then run the probe program in MDI and insert probe data such as tool length and diameter into the respective line on the Offsets page. This is option that will be used the most. Another option is to copy the code created to the clipboard.

The **second option** is to **(Probe the tool manually)**. Cursor down arrow to highlight in yellow as below. This will run similar to the VQC Manual Length only. Tool must be moved within .4" of probe.

AUTOMATIC PROBE OPTIONS	Tool Type: SHELL MILL
<ul style="list-style-type: none"> * Probe selected tool. * Probe selected tool manually. * Probe selected tool for breakage/wear. * Probe all tools. * Cancel probe selection. 	
Jog tool down within 0.40 in above the probe. Press [CYCLE START] to run probe or "[INSERT] to copy probe code to the clipboard.	

The **third option** is to **(Probe selected tool for breakage/wear)**.

<< TOOL INFO		PROBING			TOOL OFFSET >>	
IPS ON	APPROX	APPROX	EDGE	TOOL	PROB	
TOOL	LENGTH	DIAMETER	HEIG	TOLEFANC	TYPE	
1 SPINDLE	3.0000	2.0000	0.2000	0.0300	3-LEN & DIA	
2	0.	0.	0.	0.	0-NONE	
3	0.	0.	0.	0.	0-NONE	
4	0.	0.	0.	0.	0-NONE	
5	0.	0.	0.	0.	0-NONE	
6	0.	0.	0.	0.	0-NONE	
7	0.	0.	0.	0.	0-NONE	
8	0.	0.	0.	0.	0-NONE	
9	0.	0.	0.	0.	0-NONE	

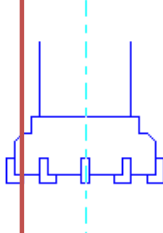
ENTER A VALUE. PRESS [WRITE] TO ADD OR [F1] TO SET THE VALUE.

AUTOMATIC PROBE OPTIONS

- * Probe selected tool.
- * Probe selected tool manually.
- * Probe selected tool for breakage/wear.
- * Probe all tools.
- * Cancel probe selection.

Press [CYCLE START] to run probe or
"[INSERT] to copy probe code to the clipboard.

Tool Type: SHELL MILL



With this selection to work a tolerance must be indicated: If a tool length is found outside of the indicated tolerance a broken tool alarm will come up on the machine. This code may be inserted into a regular program by using the [INSERT] key. This will put the code in the clipboard where it may be pasted into an individual program after a tool has been run to see if it is still functional.

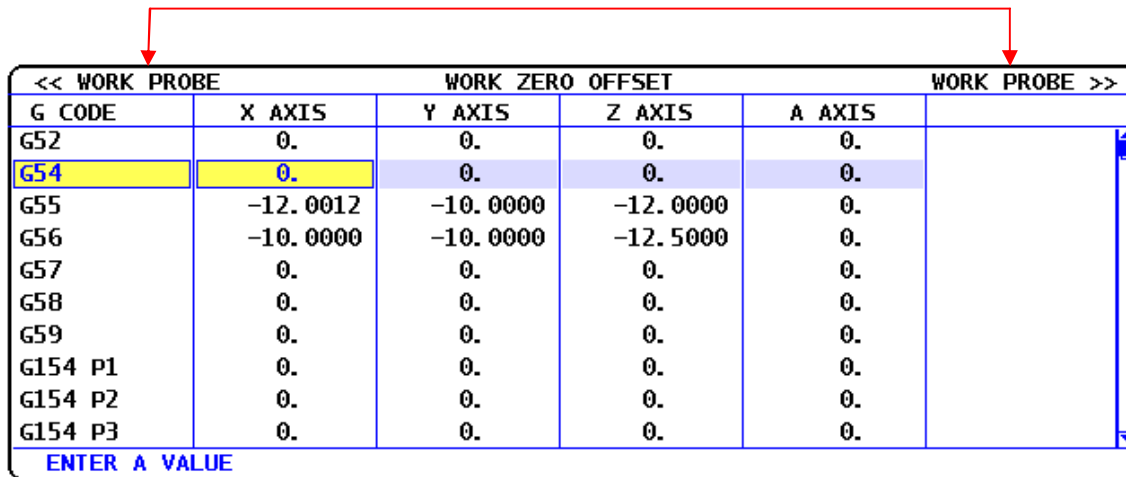
The **fourth option** is to (**Probe all tools**). If this is highlighted in yellow **all tools** which are **highlighted in green** in the PROP/TYPE column will be probed.

The **fifth option** will get out of this screen and return to the previous screen.

Setting of Work using IPS

IPS work offset probing is entered thru the work offsets page. Just as in probing for tool length offsets the machine must be in either the edit mode or preferably MDI mode. Open up the work offset page and then cursor to the work offset that is going to be determined (example G54). See below.

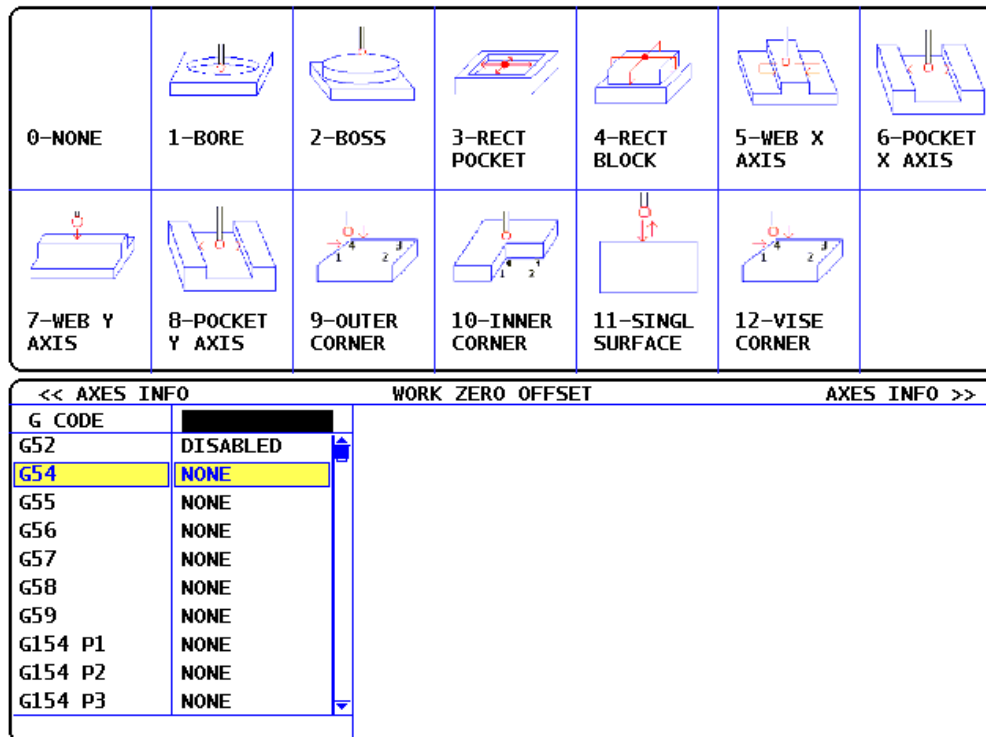
Note the <<WORK PROBE WORK PROBE>> on the top row. Using the right or left cursor key will get to the Work Probe Screen.



<< WORK PROBE		WORK ZERO OFFSET			WORK PROBE >>	
G CODE	X AXIS	Y AXIS	Z AXIS	A AXIS		
G52	0.	0.	0.	0.		
G54	0.	0.	0.	0.		
G55	-12.0012	-10.0000	-12.0000	0.		
G56	-10.0000	-10.0000	-12.5000	0.		
G57	0.	0.	0.	0.		
G58	0.	0.	0.	0.		
G59	0.	0.	0.	0.		
G154 P1	0.	0.	0.	0.		
G154 P2	0.	0.	0.	0.		
G154 P3	0.	0.	0.	0.		

ENTER A VALUE

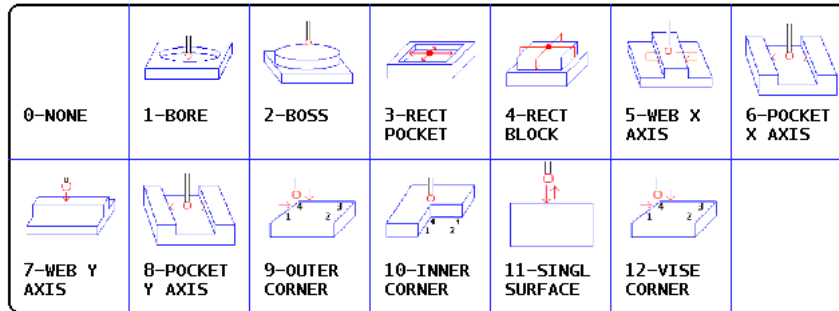
Work Offset Probe Selection Screen



<< AXES INFO		WORK ZERO OFFSET			AXES INFO >>	
G CODE						
G52	DISABLED					
G54	NONE					
G55	NONE					
G56	NONE					
G57	NONE					
G58	NONE					
G59	NONE					
G154 P1	NONE					
G154 P2	NONE					
G154 P3	NONE					

Next one of the options is selected by keying a number and pressing enter.

Below the 1-BORE option has been selected.



<< AXES INFO		WORK ZERO OFFSET		AXES INFO >>	
G CODE	PROBE ACTION	WORK PROBE INPUTS			
G52	DISABLED	Approx. Dia.	0.		
G54	BORE	X Adjust	0.		
G55	NONE	Y Adjust	0.		
G56	NONE				
G57	NONE				
G58	NONE				
G59	NONE				
G154 P1	NONE				
G154 P2	NONE				
G154 P3	NONE				

BORE

Position probe near the approximate center and just below the Z axis surface.

Enter a value larger than the approximate diameter/width of part to be probed.

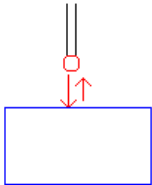
Press CYCLE START to run probe or press INSERT to copy probe code to the clipboard.

<< AXES INFO		WORK ZERO OFFSET		AXES INFO >>	
G CODE	PROBE ACTION	WORK PROBE INPUTS			
G52	DISABLED	Approx. Dia.	0.		
G54	BORE	X Adjust	0.		
G55	NONE	Y Adjust	0.		
G56	NONE				
G57	NONE				
G58	NONE				
G59	NONE				
G154 P1	NONE				
G154 P2	NONE				
G154 P3	NONE				

Right cursor opens the **work probe input box**. Inside are directions on how to position the probe and also information on which values need to be entered. The X and Y Adjust need only be used if the work offset is a certain distance away in X and Y from the center of the Bore. Follow directions and press Cycle Start to probe. In this instance a only X and Y value will be placed in the G54 work offset line.

To get the Z value the 11-Single Surface option needs to be entered.

SINGLE SURFACE



Position probe near the surface you want to measure.

Distance and direction (+/-) in the Z axis to move in incremental mode.

Press CYCLE START to run probe or press INSERT to copy probe code to the clipboard.

<< AXES INFO		WORK ZERO OFFSET		AXES INFO >>
G CODE	PROBE ACTION	WORK PROBE INPUTS		
G52	DISABLED	X Distance	0.	
G54	SINGLE SURFA	Y Distance	0.	
G55	NONE	Z distance	-0.4000	
G56	NONE			
G57	NONE			
G58	NONE			
G59	NONE			
G154 P1	NONE			
G154 P2	NONE			
G154 P3	NONE			

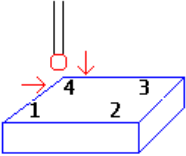
Right arrow to enter the Work Probe Input Box, select the Z distance, and enter -40. The distance and direction of the probe movement must be indicated. After running this the work offset for G54 has been set.

Note X and Y individual probe routines may also be run. The only thing to remember is that the distance and direction the probe is to be moved must be indicated inside the Work Probe Input box.

To set the X and Y value for a Vise Corner, use Option #12.

The probe needs to be moved directly above the center of the corner to be probed. Incremental distances in Z, X and Y need to be input. Running the program will give only X and Y values for corner probed. The Z value will need to be probed separately using 11- Single Surface routine.

WISE CORNER



Position probe near the approximate center and just above the Z axis surface.

Corner to be probed. 1=bottom left, 2=bottom right, 3=top right, 4=top left - default

Press CYCLE START to run probe or press INSERT to copy probe code to the clipboard.

<< AXES INFO		WORK ZERO OFFSET	AXES INFO >>
G CODE	PROBE ACTION	WORK PROBE INPUTS	
G52	DISABLED	Corner	4
G54	WISE CORNER	Incremental Z	0.4000
G55	NONE	Incremental X	1.0000
G56	NONE	Incremental Y	1.0000
G57	NONE		
G58	NONE		
G59	NONE		
G154 P1	NONE		
G154 P2	NONE		
G154 P3	NONE		